Chapter 22. Predicting Aerodynamically Generated Noise

This chapter provides an overview of ANSYS FLUENT’s approaches to computing aerodynamically generated sound, the model setup, and the procedure for computing sound. For more information about the underlying theory behind aerodynamically generated sound, see Chapter 14: Aerodynamically Generated Noise in the separate Theory Guide.

- Section 22.1: Overview
- Section 22.2: Using the Ffowcs Williams and Hawkings Acoustics Model
- Section 22.3: Using the Broadband Noise Source Models

22.1 Overview

Considering the breadth of the discipline and the challenges encountered in aerodynamically generated noise, it is not surprising that a number of computational approaches have been proposed over the years whose sophistication, applicability, and cost widely vary.

ANSYS FLUENT offers three approaches to computing aerodynamically generated noise; a direct method, an integral method based on acoustic analogy and a method that utilizes broadband noise source models.

22.1.1 Direct Method

In this method, both generation and propagation of sound waves are directly computed by solving the appropriate fluid dynamics equations. Prediction of sound waves always requires time-accurate solutions to the governing equations. Furthermore, in most practical applications of the direct method, one has to employ governing equations that are capable of modeling viscous and turbulence effects, such as unsteady Navier-Stokes equations (i.e., DNS), RANS equations, and filtered equations used in DES and LES.
The direct method is thus computationally difficult and expensive inasmuch as it requires highly accurate numerics, very fine computational meshes all the way to receivers, and acoustically nonreflecting boundary conditions. The computational cost becomes prohibitive when sound is to be predicted in the far field (e.g., hundreds of chord-lengths in the case of an airfoil). The direct method becomes feasible when receivers are in the near field (e.g., cabin noise). In many such situations involving near-field sound, sounds (or pseudo-sounds for that matter) are predominantly due to local hydrodynamic pressure which can be predicted with a reasonable cost and accuracy.

Since sound propagation is directly resolved in this method, one normally needs to solve the compressible form of the governing equations (e.g., compressible RANS equations, compressible form of filtered equations for LES). Only in situations where the flow is low subsonic and the receivers in the near field sense primarily local hydrodynamic pressure fluctuations (i.e., pseudo sound) can incompressible flow formulations be used. But this incompressible treatment will also not allow to simulate resonance and feedback phenomena.

### 22.1.2 Integral Method Based on Acoustic Analogy

For predictions of mid- to far-field noise, the methods based on Lighthill’s acoustic analogy [42] offer viable alternatives to the direct method. In this approach, the near-field flow obtained from appropriate governing equations such as unsteady RANS equations, DES, or LES are used to predict the sound with the aid of analytically derived integral solutions to wave equations. The acoustic analogy essentially decouples the propagation of sound from its generation, allowing one to separate the flow solution process from the acoustics analysis.

ANSYS FLUENT offers a method based on the Ffowcs Williams and Hawkings (FW-H) formulation [22]. The FW-H formulation adopts the most general form of Lighthill’s acoustic analogy, and is capable of predicting sound generated by equivalent acoustic sources such as monopoles, dipoles, and quadrupoles. ANSYS FLUENT adopts a time-domain integral formulation wherein time histories of sound pressure, or acoustic signals, at prescribed receiver locations are directly computed by evaluating a few surface integrals.

Time-accurate solutions of the flow-field variables, such as pressure, velocity components, and density on source (emission) surfaces, are required to evaluate the surface integrals. Time-accurate solutions can be obtained from unsteady Reynolds-averaged Navier-Stokes (URANS) equations, large eddy simulation (LES), or detached eddy simulation (DES) as appropriate for the flow at hand and the features that you want to capture (e.g., vortex shedding). The source surfaces can be placed not only on impermeable walls, but also on interior (permeable) surfaces, which enables you to account for the contributions from the quadrupoles enclosed by the source surfaces. Both broadband and tonal noise can be predicted depending on the nature of the flow (noise source) being considered, turbulence model employed, and the time scale of the flow resolved in the flow calculation.
The FW-H acoustics model in ANSYS FLUENT allows you to select multiple source surfaces and receivers. It also permits you either to save the source data for a future use, or to carry out an “on the fly” acoustic calculation simultaneously as the transient flow calculation proceeds, or both. Sound pressure signals thus obtained can be processed using the fast Fourier transform (FFT) and associated postprocessing capabilities to compute and plot such acoustic quantities as the overall sound pressure level (SPL) and power spectra.

One important limitation of ANSYS FLUENT’s FW-H model is that it is applicable only to predicting the propagation of sound toward free space. Thus, while the model can be legitimately used to predict far-field noise due to external aerodynamic flows, such as the flows around ground vehicles and aircrafts, it cannot be used for predicting the noise propagation inside ducts or wall-enclosed space.

### 22.1.3 Broadband Noise Source Models

In many practical applications involving turbulent flows, noise does not have any distinct tones, and the sound energy is continuously distributed over a broad range of frequencies. In those situations involving broadband noise, statistical turbulence quantities readily computable from RANS equations can be utilized, in conjunction with semi-empirical correlations and Lighthill’s acoustic analogy, to shed some light on the source of broadband noise.

ANSYS FLUENT offers several such source models that enable you to quantify the local contribution (per unit surface area or volume) to the total acoustic power generated by the flow. They include the following:

- Proudman’s formula
- jet noise source model
- boundary layer noise source model
- source terms in the linearized Euler equations
- source terms in Lilley’s equation

Considering that one would ultimately want to come up with some measures to mitigate the noise generated by the flow in question, the source models can be employed to extract useful diagnostics on the noise source to determine which portion of the flow is primarily responsible for the noise generation. Note, however, that these source models do not predict the sound at receivers.
Unlike the direct method and the FW-H integral method, the broadband noise source models do not require transient solutions to any governing fluid dynamics equations. All the source models need is what typical RANS models would provide, such as the mean velocity field, turbulent kinetic energy ($k$) and the dissipation rate ($\varepsilon$). Therefore, the use of broadband noise source models requires the least computational resources.

### 22.2 Using the Ffowcs Williams and Hawkings Acoustics Model

The procedure for computing sound using the FW-H acoustics model in ANSYS FLUENT consists largely of two steps. In the first step, a time-accurate flow solution is generated from which time histories of the relevant variables (e.g., pressure, velocity, and density) on the selected source surfaces are obtained. In the second step, sound pressure signals at the user-specified receiver locations are computed using the source data collected during the first step.

Note that you can also use the FW-H model for a steady-state simulation in the case where your model has a single rotating reference frame. Here, the thickness and loading noise due to the motion of the noise sources is computed using the FW-H integrals (see Equation 14.2-5 and Equation 14.2-6 in the separate Theory Guide), except that the term involving the time derivative of surface pressure (contribution to $\dot{L}_r$ in Equation 14.2-6 in the separate Theory Guide) is set to zero.

In computing sound pressure using the FW-H integral solution, ANSYS FLUENT uses a so-called “forward-time projection” to account for the time delay between the emission time (the time at which the sound is emitted from the source) and the reception time (the time at which the sound arrives at the receiver location). The forward-time projection approach enables you to compute sound at the same time “on the fly” as the transient flow solution progresses, without having to save the source data.

In this section, the procedure for setting up and using the FW-H acoustics model is outlined first, followed by detailed descriptions of each of the steps involved. Remember that only the steps that are pertinent to acoustics modeling are discussed here. For information about the inputs related to other models that you are using in conjunction with the FW-H acoustics model, see the appropriate sections for those models.
22.2 Using the Ffowcs Williams and Hawkings Acoustics Model

The general procedure for carrying out an FW-H acoustics calculation in ANSYS FLUENT is as follows:

1. Calculate a converged flow solution. For a transient case, run the transient solution until you obtain a “statistically steady-state” solution as described below.

2. Enable the FW-H acoustics model and set the associated model parameters.
   - ![Models → Acoustics → Edit...](#)

3. Specify the source surface(s) and choose the options associated with acquisition and saving of the source data. For a steady-state case, specify the rotating surface zone(s) as the source surface(s).

4. Specify the receiver location(s).

5. Continue the transient solution for a sufficiently long period of time and save the source data (transient cases only).
   - ![Run Calculation](#)

6. Compute and save the sound pressure signals.
   - ![Run Calculation → Acoustic Signals...](#)

7. Postprocess the sound pressure signals.
   - ![Plots → FFT → Set Up...](#)

Before you start the acoustics calculation for a transient case, an ANSYS FLUENT transient solution should have been run to a point where the transient flow field has become “statistically steady”. In practice, this means that the unsteady flow field under consideration, including all the major flow variables, has become fully developed in such a way that its statistics do not change with time. Monitoring the major flow variables at selected points in the domain is helpful for determining if this condition has been met.

As discussed earlier, URANS, DES, and LES are all legitimate candidates for transient flow calculations. For stationary source surfaces, the frequency of the aerodynamically generated sound heard at the receivers is largely determined by the time scale or frequency of the underlying flow. Therefore, one way to determine the time-step size for the transient computation is to make it small enough to resolve the smallest characteristic time scale of the flow at hand that can be reproduced by the mesh and turbulence adopted in your model.

Once you have obtained a statistically stationary flow-field solution, you are ready to acquire the source data.
22.2.1 Enabling the FW-H Acoustics Model

To enable the FW-H acoustics model, select Ffowcs-Williams & Hawkings in the Acoustics Model dialog box (Figure 22.2.1).

When you select Ffowcs-Williams & Hawkings, the dialog box will expand to show the relevant fields for user inputs.

Figure 22.2.1: The Acoustics Model Dialog Box

Setting Model Constants

Under Model Constants in the Acoustics Model dialog box, specify the relevant acoustic parameters and constants used by the model.

- **Far-Field Density** (for example, \( \rho_0 \) in Equation 14.2-1 in the separate Theory Guide) is the far-field fluid density.

- **Far-Field Sound Speed** (for example, \( a_0 \) in Equation 14.2-1) is the sound speed in the far field (= \( \sqrt{\gamma RT_0} \)).

- **Reference Acoustic Pressure** (for example, \( p_{\text{ref}} \) in Equation 29.11-11) is used to calculate the sound pressure level in dB (see Section 29.11.4: Using the FFT Utility). The default reference acoustic pressure is \( 2 \times 10^{-5} \) Pa.

- **Number of Time Steps Per Revolution** is available only for steady-state cases that have a single rotating reference frame. Here you will specify the number of equivalent time steps that it will take for the rotating zone to complete one revolution.
22.2 Using the Ffowcs Williams and Hawkings Acoustics Model

**Number of Revolutions** is available only for steady-state cases that have a single rotating reference frame. Here you will specify the number of revolutions that will be simulated in the model.

**Source Correlation Length** is required when sound is to be computed using a 2D flow result. The FW-H integrals will be evaluated over this length in the depth-wise direction using the identical source data (see Figure 22.2.2).

The default values are appropriate for sound propagating in air at atmospheric pressure and temperature.

**Computing Sound “on the Fly”**

The FW-H acoustics model in ANSYS FLUENT allows you to perform simultaneous calculation of the sound pressure signals at the prescribed receivers without having to write the source data to files, which can save a significant amount of disk space on your machine. To enable this “on-the-fly” calculation of sound, enable the Compute Acoustic Signals Simultaneously option in the Acoustics Model dialog box.

Because the noise computation takes a negligible percentage of memory and computational time compared to a transient flow calculation, this option can be used by itself or along with the process of source data file export and sound calculation. For the latter, computing signals “on the fly” allows you to see when the signals have become statistically steady so you can know when to stop the simulation.

When the Compute Acoustic Signals Simultaneously option is enabled, the ANSYS FLUENT console window will print a message at the end of each time step indicating that the sound pressure signals have been computed (e.g., Computing sound signals at x receiver locations ..., where x is the number of receivers you specified). Enabling this option instructs ANSYS FLUENT to compute sound pressure signals at the end of each time step, which will slightly increase the computation time.

Note that this option is available only when the FW-H acoustics model has been enabled. See below for details about exporting source data without enabling the FW-H model.
Writing Source Data Files

Although the “on-the-fly” capability is a convenient feature, you will want to save the source data as well, because the acquisition of source data during a transient flow-field calculation is the most time-consuming part of acoustics computations, and you most likely will not want to discard it. By saving the source data, you can always reuse it to compute the sound pressure signals at new or additional receiver locations.

To save the source data to files, enable either the Export Acoustic Source Data in ASD Format or the Export Acoustic Source Data in CGNS Format option, or both. After you have made your selection, the relevant source data at all face elements of the selected source surfaces will be written into the files you specify. The source data vary depending on the solver option you have chosen and whether the source surface is a wall or not. Table 22.2.1 shows the flow variables saved as the source data.

Figure 22.2.2: The Acoustics Model Dialog Box for a 2D Steady-State Case with a Single Rotating Reference Frame

See Section 22.2.2: Specifying Source Surfaces for details on how to specify parameters for exporting source data.
### Table 22.2.1: Source Data Saved in Source Data Files

<table>
<thead>
<tr>
<th>Solver Option</th>
<th>Source Surface</th>
<th>Source Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>incompressible walls</td>
<td>$p$</td>
<td></td>
</tr>
<tr>
<td>incompressible permeable surfaces</td>
<td>$p, u, v, w$</td>
<td></td>
</tr>
<tr>
<td>compressible walls</td>
<td>$p$</td>
<td></td>
</tr>
<tr>
<td>compressible permeable surfaces</td>
<td>$\rho, p, u, v, w$</td>
<td></td>
</tr>
</tbody>
</table>

### Exporting Source Data Without Enabling the FW-H Model: Using the ANSYS FLUENT ASD Format

You can export sound source data for use with SYSNOISE without having to enable the Ffowcs Williams and Hawkings (FW-H) model. You will still need to specify source surfaces (see Section 22.2.2: Specifying Source Surfaces), as `.index` and `.asd` files are required by SYSNOISE. In addition, you can choose fluid zones as emission sources if you want to export quadrupole sources. To enable the selection of fluid zones as sources, use the

```
define → models → acoustics → export-volumetric-sources?
```

text command and change the selection to `yes`.

SYSNOISE also requires centroid data for source zones that are being exported.

For fan noise calculations, once you have specified the source zones in the Acoustic Sources dialog box and you have selected Export Acoustic Source Data in ASD Format from the Acoustics Model dialog box, you can export geometry in cylindrical coordinates by using the

```
define → models → acoustics → cylindrical-export?
```

text command and changing the selection to `yes`. By default, ANSYS FLUENT exports source zones for SYSNOISE in Cartesian coordinates.

You can then export the centroid data to a data file using the following text command:

```
define → models → acoustics → write-centroid-info
```

Since you will not be using the FW-H model to compute signals, you will not need to specify any acoustic model parameters or receiver locations. Also, you will not be able to enable the Compute Acoustic Signals Simultaneously option in the Acoustics Model dialog box, and Acoustic Signals... will not be available in the Run Calculation task page.
Exporting Source Data Without Enabling the FW-H Model: Using the CGNS Format

The sound source data for non-permeable surfaces can be exported in the CGNS file format (for Virtual Lab) without having to enable the Ffowcs Williams and Hawkings (FW-H) model. Enable the Export Acoustic Source Data in CGNS Format option in the Acoustics Model dialog box (Figure 22.2.3). Specify the source surfaces in the Acoustics Sources dialog box (see Section 22.2.2: Specifying Source Surfaces) where, by default, the Number of Time Steps per File is set to 1.

![Acoustics Model Dialog Box for Exporting in CGNS Format](image)

Virtual Lab requires a mesh data file (named `<prefix>_mesh.cgns`) and a solution data file (named `<prefix>_<timestep>.cgns`). The string `<prefix>` is a generic name, which you will specify in the Source Data Root File Name in the Acoustics Sources dialog box. There is one single solution data file (.cgns) per time level exported, which contains the static pressure at the wall-face centroid location. The .cgns files will be stored in a directory, which you specify (named `<directory_name>/<prefix>`) in the Source Data Root File Name.

In addition, you can export quadrupole sources data by choosing fluid zones as emission sources. To enable the selection of fluid zones as sources, use the text command:

```
define→models→acoustics→export-volumetric-sources-cgns?
```

When asked if you would like to Export volumetric sources? enter yes. Note that Virtual Lab requires volumetric mesh data file (`<prefix>_Q_mesh.cgns`) and quadrupole solution data files (`<prefix>_Q_<timestep>.cgns`). The .cgns file will be stored in a similar way to that of dipole data export, in the directory specified by you in the Source Data Root File Name text entry box.
22.2 Using the Ffowcs Williams and Hawkings Acoustics Model

22.2.2 Specifying Source Surfaces

In the Acoustics Model dialog box, click the Define Sources... button to open the Acoustic Sources dialog box (Figure 22.2.4). Here you will specify the source surface(s) to be used in the acoustics calculation and the inputs associated with saving source data to files.

Under Source Zones, you can select multiple emission (source) surfaces and the surface Type that you can select is not limited to a wall. You can also choose interior surfaces and sliding interfaces (both stationary and rotating) as source surfaces.

The ability to choose multiple source surfaces is useful for investigating the contributions from individual source surfaces. The results based on the use of multiple source surfaces are valid as long as there are negligible acoustic interactions among the surfaces. Thus, some caution needs to be taken when selecting multiple source surfaces.
In cases where multiple source surfaces are selected, no source surface may enclose any of the other source surfaces. Otherwise, the sound pressure calculated based on the source surfaces will not be accurate, as the contribution from the enclosed (inner) source surfaces is over predicted, since the FW-H model is unable to account for the shading of the sound from the inner source surfaces by the enclosure surface.

If you specify any interior surfaces as source surfaces, the interior surface must be generated in advance (e.g., in GAMBIT) in such a way that the two cell zones adjacent to the surface have different cell zone IDs. Furthermore, you must correctly specify which of the two zones is occupied by the quadrupole sources (interior cell zone). This will allow ANSYS FLUENT to determine the direction in which the sound will propagate. When you first attempt to select a legitimate interior surface (i.e., an interior surface having two different cell zones on both sides) as a source surface, the Interior Cell Zone Selection dialog box (Figure 22.2.5) will appear. You will then need to select the interior cell zones from the two zones listed under the Interior Cell Zone. Figure 22.2.6 shows an example of an interior source surface.

![Image of Interior Cell Zone Selection Dialog Box](image)

**Figure 22.2.5**: The Interior Cell Zone Selection Dialog Box

Like general interior surfaces, if the source surfaces selected are sliding interfaces, a dialog box similar to Figure 22.2.5 will appear that will show the two adjacent cell zones and you will be asked to specify the zone which has the sound sources.

When a permeable surface (either interior or sliding interface) is chosen as the source surface, other wall surfaces inside the volume enclosed by the permeable surface that generate sound should not be chosen for the acoustics calculation. For example, when running an “on-the-fly” calculation, if both these surfaces are selected, the sound pressure will be counted twice.
22.2 Using the Ffowcs Williams and Hawkings Acoustics Model

**Figure 22.2.6: An Interior Source Surface**

**Saving Source Data**

To save the source data, you have to specify the Source Data Root File Name, Write Frequency (in number of time steps), and Number of Time Steps per File in the Acoustic Sources dialog box.

The Source Data Root File Name is used to give the names of the source data files (e.g., acoustic_examplexxxx.asd, where xxxx is the global time-step index of the transient solution) and an index file (e.g., acoustic_example.index) that will store the information associated with the source data. The Write Frequency allows you to control how often the source data will be written. This will enable you to save disk space if the time-step size used in the transient flow simulation is smaller than necessary to resolve the sound frequency you are attempting to predict. In most situations, however, you will want to save the source data at every time step and use the default value of 1.

Since acoustics calculations usually generate thousands of time steps of source data, you may want to split the data into several files. Specifying the Number of Time Steps per File allows you to write the source data into separate files for different simulation intervals, the duration of which (in terms of the number of transient flow time steps) is specified by you. For example, if you specify 100 for this parameter, each file will contain source data for an interval length of 100 time steps regardless of the write frequency.
You will find this feature useful if you want to use a selected number of source data files to compute the sound pressure rather than using all the data. For example, you may want to exclude an initial portion of the source data from your acoustics calculation because you may realize later that the flow field has not fully attained a statistically steady state.

After you click Apply, ANSYS FLUENT will create the index file (e.g., `acoustics_example.index`), which contains information about the source data.

If you choose to save source data, keep in mind that the source data can use up a considerable amount of disk space, especially if the mesh being used has a large number of face elements on the source surfaces you selected. ANSYS FLUENT will print out the disk space requirement per time step at the time of source surface selection if the Export Acoustic Source Data in ASD Format option is enabled in the Acoustics Model dialog box.

At this point, if you have chosen to perform your acoustics calculation in two steps, (i.e., saving the source data first, and computing the sound at a later time), you can go ahead and instruct ANSYS FLUENT to perform a suitable number of time steps, and the source data will be saved to the disk. If you have chosen to perform an “on-the-fly” acoustic calculation, then you will need to specify receiver locations (see Section 22.2.3: Specifying Acoustic Receivers) before you run the unsteady ANSYS FLUENT solution any further.

### 22.2.3 Specifying Acoustic Receivers

In the Acoustics Model dialog box, click the Define Receivers... button to open the Acoustic Receivers dialog box (Figure 22.2.7).

![Acoustic Receivers Dialog Box](image)

Figure 22.2.7: The Acoustic Receivers Dialog Box

Note that you can also open the Acoustic Receivers dialog box by clicking the Receivers... button in the Acoustic Sources or the Acoustic Signals dialog box.
Increase the **Number of Receivers** to the total number of receivers for which you want to compute sound, and enter the coordinates for each receiver in the X-Coord., Y-Coord., and Z-Coord. fields. Note that because ANSYS FLUENT’s acoustics model is ideally suited for far-field noise prediction, the receiver locations you define should be at a reasonable distance from the sources of sound (i.e., the selected source surfaces). The receiver locations can also fall outside of the computational domain.

For each receiver, you can specify a file name in the **Signal File Name** field. These files will be used to store the sound pressure signals at the corresponding receivers. By default, the files will be named `receiver-1.ard`, `receiver-2.ard`, etc.

Once the receiver locations have been defined, the setup for your acoustic calculation is complete.

### 22.2.4 Specifying the Time Step

When using an implicit-in-time solution algorithm (dual-time stepping), and depending on the physical time step size and the most important time scales in the flow, you can write the acoustic source data at every time step. You can also coarsen it in time by a given frequency factor. The highest possible frequency the acoustic analysis can generate is based on the time step size of the collected acoustic source data.

When using the density-based explicit solver, the physical time step must be computed by the solver, based on the CFL condition (Courant number). Due to the possibly large fluctuations of the physical time step, an adapting time-stepping procedure can be used when the FW-H acoustics model is enabled. This allows you to use a user-specified time interval for sampling the acoustic data. In turn, the solver adapts its time step, when necessary, without violating the CFL conditions to make sure that data is available at the desired time interval (hence, avoiding data interpolations).

In the **Run Calculation** task page (Figure 22.2.8), enter the **Time Step Size for Acoustic Data Export** to specify the time interval for acoustic data sampling. The value of this constant time step size determines the highest frequency that the acoustic analysis reproduces.

You can refer to Section 26.12: Performing Time-Dependent Calculations for more information about the Run Calculation task page.

#### Run Calculation

You can now proceed to instruct ANSYS FLUENT to perform a transient calculation for a suitable number of time steps. When the calculation is finished, you will have either the source data saved on files (if you chose to save it to a file or files), or the sound pressure signals (if you chose to perform an acoustic calculation “on the fly”), or both (if you chose to save the source data to files *and* if you chose to perform the acoustic calculation “on the fly”).
Figure 22.2.8: The Run Calculation Task Page
If you chose to save the source data to files, the ANSYS FLUENT console window will print a message at the end of each time step indicating that source data have been written (or appended to) a file (e.g., `acoustic_example240.asd`).

### 22.2.5 Postprocessing the FW-H Acoustics Model Data

At this point, you will have either the source data saved to files or the sound pressure signals computed, or both. You can process these data to compute and plot various acoustic quantities using ANSYS FLUENT's FFT capabilities. See Section 29.11: Fast Fourier Transform (FFT) Postprocessing for more information.

#### Writing Acoustic Signals

If you chose to perform the acoustic calculation “on the fly”, you will need to write the sound pressure data to files. To do so, select **Write Acoustic Signals** under **Options** in the **Acoustic Signals** dialog box (Figure 22.2.9) and then click **Write**. The computed acoustic pressure will be saved from internal buffer memory into a separate file for each receiver you defined in the **Acoustic Receivers** dialog box (e.g., `receiver-1.ard`).

- **Run Calculation** → **Acoustic Signals**...

#### Reading Unsteady Acoustic Source Data

Computing the sound pressure signals using the source data saved to files is done in the **Acoustic Signals** dialog box (Figure 22.2.9)

- **Run Calculation** → **Acoustic Signals**...

To compute the sound data, use the following procedure:

1. In the **Acoustic Signals** dialog box, select **Read Unsteady Acoustic Source Data Files** under **Options**.

2. Click **Load Index File...** and select the index file for your computation in the **Select File** dialog box. The file will have the name you entered in the **Source Data Root File Name** field in the **Acoustic Sources** dialog box, followed by the .index suffix (e.g., `acoustic_example.index`).

3. In the **Source Data Files** list, select the source data files that you want to use to compute sound. Source data files will all contain the specified root file name followed by the suffix .asd.

   - **You can use any number of source data files. However, note that you should select only consecutive files.**
4. In the **Active Source Zones** list, select the source zones you want to include to compute sound. See Section **22.2.2: Specifying Source Surfaces** for details about proper source surface selection.

5. In the **Receivers** list, select the receivers for which you want to compute and save sound.
   
   Optionally, you can click the **Receivers...** button to open the **Acoustic Receivers** dialog box and define additional receivers.

6. Click the **Compute/Write** button to compute and save the sound pressure data. One file will be saved for each receiver you previously specified in the **Acoustic Receivers** dialog box (e.g., `receiver-1.ard`).
If you enabled both the **Export Acoustic Source Data in ASD Format** and **Compute Acoustic Signals Simultaneously** options in the Acoustics Model dialog box, you will need to first select the **Write Acoustic Signals** option in the Acoustic Signals dialog box after the flow simulation has been completed.

If you select the **Read Unsteady Acoustic Source Data Files** before writing out the “on-the-fly” data in such a case, the data will be flushed out of the internal buffer memory. To avoid such a loss of data, you should save the ANSYS FLUENT case and data files whenever you begin to do an acoustic computation in the Acoustic Signals dialog box. The sound pressure data calculated “on the fly” will then be saved into the `.dat` file. Finally, after the “on-the-fly” data is saved, make sure to change the file names of the receivers before doing a sound pressure calculation with the **Read Unsteady Acoustic Source Data Files** option enabled, to avoid overwriting the “on-the-fly” signal files.

Note that you can compute and write sound pressure signals only when the FW-H acoustics model has been enabled. See Section 22.2.1: Exporting Source Data Without Enabling the FW-H Model: Using the ANSYS FLUENT ASD Format for details about exporting source data (e.g., for SYSNOISE) without enabling the FW-H model.

**Pruning the Signal Data Automatically**

Before the computed sound pressure data at each receiver is saved, it is by default automatically pruned. Pruning of the receiver data means clipping the tails of the signal where incomplete source information is available.

The acoustic source data is tabulated from time $\tau_0$ to $\tau_n$. Without auto-pruning, the receiver register begins receiving the earliest sound pressure signal at

$$t_0 = \tau_0 + \frac{r_{\text{min}}}{a_0}$$

where $r_{\text{min}}$ is the shortest distance between the source surfaces and the receiver. However, the receiver will not receive the sound pressure signal from the farthest point on the source surfaces ($r_{\text{max}}$) until the receiver time becomes

$$t_1 = \tau_0 + \frac{r_{\text{max}}}{a_0}$$
From time $t_0$ to $t_1$, the sound accumulated on the receiver register does not include the contribution from the entire source surface area, and thus the sound pressure data received during that time is not complete. The same thing occurs during the period from

$$t_m = \tau_m + \frac{r_{\text{min}}}{a_0}$$

to

$$t_n = \tau_n + \frac{r_{\text{max}}}{a_0}$$

Thus, pruning means clipping the signal on the incomplete ends, from $t_0$ to $t_1$ and $t_m$ to $t_n$. Auto-pruning can be disabled using the `define models acoustics auto-prune` text command. Although auto-pruning can be disabled, it is expected that you will use only the complete sound pressure data.

**Reporting the Static Pressure Time Derivative**

The RMS value of the static pressure time derivative ($\partial p/\partial t$) is available for postprocessing only on wall surfaces, which are at the same time sources of sound, when the FW-H acoustics model is used.

You can select `Surface dpdt RMS` in the `Acoustics...` category only when you specify at least one wall surface, which is also marked as an acoustic source, in the relevant postprocessing dialog boxes.

**Using the FFT Capabilities**

Once the sound pressure signals are computed and saved in files, the sound data is ready to be analyzed using ANSYS FLUENT’s FFT tools. In the Fourier Transform dialog box (Figure 29.11.1), click on `Load Input File...` and select the appropriate `.ard` file. If the receiver data is still in ANSYS FLUENT’s memory, then it can directly be processed using the `Process Receiver` option. See Section 29.11: Fast Fourier Transform (FFT) Postprocessing for more information on ANSYS FLUENT’s FFT capabilities.
22.3 Using the Broadband Noise Source Models

In this section, the procedure for setting up and using the broadband noise source models is outlined first, followed by descriptions of each of the steps involved.

The general procedure for carrying out a broadband noise source calculation in ANSYS FLUENT is as follows:

1. Calculate a steady or unsteady RANS solution.
2. Enable the broadband noise model and set the associated model parameters.

   ![Models] Acoustics Edit...

3. Postprocess the noise sources.

   ![Graphics and Animations] Contours Set Up...

22.3.1 Enabling the Broadband Noise Source Models

To enable the broadband noise sources models, select Broadband Noise Sources in the Acoustics Model dialog box (Figure 22.3.1).

![Models] Acoustics Edit...

Setting Model Constants

Under Model Constants in the Acoustics Model dialog box, specify the relevant acoustic parameters and constants used by the model. See Section 22.2.1: Enabling the FW-H Acoustics Model for the definitions of Far-Field Density and Far-Field Sound Speed.

Reference Acoustic Power (for example, $P_{ref}$ in Equation 14.2-13 in the separate Theory Guide) is used to compute the acoustic power outputs in decibels (dB). The default value is $10^{-12}$. Note that the units for the reference acoustic power will be different in 2D ($W/m^2$) and 3D ($W/m^3$) cases.

Number of Realizations is the number of samples used in the SNGR to compute the averaged source terms of LEE and Lilley’s equations. The default value is 200.

Number of Fourier Modes ($N$ in Equation 14.2-33 in the separate Theory Guide) is the number of the Fourier modes used to compute the turbulent velocity field and its derivatives. The turbulent velocity field is then used to compute the LEE and Lilley’s source terms. The default value is 50.
Figure 22.3.1: The Acoustics Model Dialog Box for Broadband Noise
22.3.2 Postprocessing the Broadband Noise Source Model Data

The final step in the broadband noise source modeling process is the postprocessing of acoustic power and noise source data. The following variables are available in the Acoustics... postprocessing category:

- Acoustic Power Level (dB)
- Acoustic Power
- Jet Acoustic Power Level (dB) (axisymmetric models only)
- Jet Acoustic Power (axisymmetric models only)
- Surface Acoustic Power Level (dB)
- Surface Acoustic Power
- Lilley’s Self-Noise Source
- Lilley’s Shear-Noise Source
- Lilley’s Total Noise Source
- LEE Self-Noise X-Source
- LEE Shear-Noise X-Source
- LEE Total Noise X-Source
- LEE Self-Noise Y-Source
- LEE Shear-Noise Y-Source
- LEE Total Noise Y-Source
- LEE Self-Noise Z-Source (3D models only)
- LEE Shear-Noise Z-Source (3D models only)
- LEE Total Noise Z-Source (3D models only)